Astrophysical Exercises
Hydrodynamics in One Dimension

Joachim Köppen Strasbourg 2006

http://astro.u-strasbg.fr/~koppen/JKHome.html
koppen@astro.u-strasbg.fr

1 Physical Background

Most of the matter in the universe, i.e. stars and interstellar matter, is in the form of gas. In comparison with our atmosphere, this gas has rather low densities: 10^6 atoms per m^3 in the dilute interstellar medium, 10^9 to 10^{11} in gas clouds, and 10^{17} to 10^{22} in the atmospheres of stars. Even at these low densities, the particles interact sufficiently often and quickly to allow a description of the state of the gas by defintion of (mass) density, pressure, temperature, and velocity of a particular volume element. The thermal behaviour of the gas then can well be described by the ideal gas equation, its dynamical behaviour by the equations of fluid dynamics. In this way, one is able to compute the evolution with time of the blast wave of a supernova explosion in the interstellar medium, the expulsion and expansion of planetary nebulae and novae, the collapse of interstellar gas clouds under the influence of its own gravity, and other phenomena.

As an example of how one approaches such a problem, we shall look at the evolution of gas enclosed in a tube. This tube will be assumed to be linear, thus we have to tackle only a one-dimensional problem. Assuming the gas to be isothermal, we avoid solving the energy equation. The latter could be rather complex, if one takes into account all the atomic processes. Though this might be necessary when modelling observations, for many studies of the time evolution of the gas it is reasonable to use the isothermal approximation. Despite these simplifications, the program will simulate the dynamics of the gas well enough, but also show the limitations and typical problems associated with the numerical solution of partial differential equations.

Apart from the linear shock tube, the program can easily be extended to compute the collisions of interstellar clouds, expansion of planetary nebulae and nova shells, and even the collapse of proto-stars and -galaxies.

2 The Equations

2.1 Conservation of Mass

The basic fluid dynamic equations come from the conservation laws of matter (i.e. mass), momentum, and energy. The first one gives the equation of continuity for the mass density ρ of gas particles flowing with a velocity $\mathbf{v}$ (vector):

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \mathbf{v}) = 0 \tag{1}$$

the right hand side is zero, since nowhere matter is created or destroyed. The first term is the time change of density in a particular volume element, the second ("advection") term decribes how this density is changed by the flow of matter into or out of this volume element. In the one-dimensional case, the velocity $v = |\mathbf{v}|$ is taken along the spatial coordinate (say x), and one gets

$$\nabla(\rho v) = \frac{1}{x^n}\frac{\partial(x^n \rho v)}{\partial x} = \frac{\partial(\rho v)}{\partial x} + n\frac{\rho v}{x} \tag{2}$$

The constant n indicates the curvature of the geometrical configuration of the problem. One has $n = 0$ for the planar or linear case, $n = 1$ for the system being cylindrically symmetric — where x corresponds to the distance from the rotational axis — and $n = 2$ for a spherically symmetric system, where x is the distance from the centre. For the linear tube $n = 0$ we thus get the first equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho v)}{\partial x} = 0 \tag{3}$$

2.2 Conservation of Momentum

This describes the changes of momentum due to the action of the action of external forces. Here we shall consider only the force due to the thermal pressure gradient $\mathbf{F} = -\nabla p$

$$\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla(\rho \mathbf{v}\mathbf{v}) = \mathbf{F} = -\nabla p \tag{4}$$

Assuming the equation of state of ideal gas

$$p = \rho\frac{kT}{m} = \rho c^2 \tag{5}$$

with the isothermal speed of sound c, and again with one dimensional geometry, one obtains

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho v^2)}{\partial x} + n\frac{\rho v^2}{x} = -c^2\frac{\partial \rho}{\partial x} \tag{6}$$

which in planar geometry reduces to

$$\frac{\partial(\rho v)}{\partial t} = -\frac{\partial(\rho v^2)}{\partial x} - c^2\frac{\partial \rho}{\partial x} = -\frac{\partial \rho(v^2 + c^2)}{\partial x} \tag{7}$$

2.3 Conservation of Energy

This gives the change of the energy density (thermal plus kinetic): $e = \frac{3}{2}c^2 + \frac{1}{2}v^2$ due to compression or expansion of the gas ($p\nabla\mathbf{v}$), and to all local heating H and cooling C processes, such as absorption and emission of photons by the gas:

$$\frac{\partial e}{\partial t} + \nabla(e\mathbf{v}) = -p\nabla\mathbf{v} + H - C \tag{8}$$

In this exercise, we shall consider two simple cases: (a) suppose that the gas has the same temperature – i.e. sound speed – everywhere. Then we do not need to solve the energy

equation. (b) suppose that the gas is adiabatic, so wee suppose that there are no heating or cooling processes ($H = C = 0$). So we solve the additional energy equation which has a simple form. These two cases are useful approximations of a real system (which could be rather complex due to the various atomic processes) and they give some essential understanding of the physics in astrophysical objects.

3 Discretization of the Equations

3.1 Principles

In the numerical method of solution of the differential equations one cannot find the functional form of the solution, but only the values of this function at a number of discrete points in space and time. Likewise, all functions describing physical parameters, such as the temperature, and the initial and boundary conditions as well, are also given only at these grid points. The selection of the proper grids may be quite essential for the stability of the solution procedure, i.e. on how well the results represent the true solution. Obviously, any structures smaller than the grid spacing will be lost in the results. But making the grids finer implies that the number of computations increases and so does the computing time.

For our problem it is quite natural to use the following approach: At each time the densities and velocities are known at each point of a fixed spatial grid. From this information, one computes how the densities and velocities change during the next interval of time, and then updates the "old" physical quantities by the "new" ones. This procedure is carried out as long as one needs.

Let us first consider the discretization in space: The region of interest (from x_{min} to x_{max}) is divided into n intervals of equal length (x_i, x_{i+1}) for $i = 1...n-1$. The values of density and velocity at any time t are taken at the interval borders, i.e. $\rho_i(t), v_i(t)$. To compute the spatial derivatives, we may employ various schemes, such as first-order approximations

$$\left.\frac{df}{dx}\right|_i = \frac{f(x_{i+1}) - f(x_i)}{x_{i+1} - x_i} \tag{9}$$

or likewise

$$\left.\frac{df}{dx}\right|_i = \frac{f(x_i) - f(x_{i-1})}{x_i - x_{i-1}} \tag{10}$$

or a centered second-order scheme

$$\left.\frac{df}{dx}\right|_i = \frac{f(x_{i+1}) - f(x_{i-1})}{x_{i+1} - x_{i-1}} \tag{11}$$

Next, we must decide on how to do the time-stepping, i.e. how to integrate the equation

$$\frac{df}{dt} = A(t) \tag{12}$$

from time t to a later time $t+\Delta t$. Among the discrete expressions for the derivative at time t we may choose the simplest ones, which are of first order. There are two possibilities: Firstly, one may chose the backward differences

$$\frac{df}{dt} = \frac{f(t) - f(t-\Delta t)}{\Delta t} = A(t) \tag{13}$$

which yields

$$f(t+\Delta t) = f(t) + A(t+\Delta t)\Delta t \tag{14}$$

This formulation is called "implicit"; it requires knowledge of the derivative A at the new time $t+\Delta t$, i.e. of the solution that has yet to be computed. This method can be used to build programs which allow quite large time steps Δt, but it may require an iterative procedure and makes the program more complex. Thus extensions are more difficult to put in.

Taking the forward differences instead

$$\frac{df}{dt} = \frac{f(t+\Delta t) - f(t)}{\Delta t} = A(t) \tag{15}$$

gives

$$f(t+\Delta t) = f(t) + A(t)\Delta t \tag{16}$$

This is called "explicit"; it requires knowledge about A only at the past time t. Hence the program can be directly written as Eqn. 16. However, it is not stable for large time steps, i.e. the numerical solution can deviate from the real solution very strongly. But if care is taken to avoid large timesteps, or sufficiently small ones are taken, this method gives reliable results, and the program can be written in a straightforward way that makes it easy to test as well as to modify it to include more effects and processes.

In principle, there is no reason why one should restrict oneself to a first-order representation. Higher order schemes have a higher possible accuracy, however they are more complicated to program, and may also be more susceptible to rapid changes in the variables, which may well cause the solution to tend to oscillate or go to wrong values at some grid points. E.g. the second-order scheme for the spatial discretization Eqn. 11 computes the changes at one point from the neighbouring points only. Thus in the solutions there is a tendency for neighbouring points to deviate alternatingly about the average density profile. This can be observed when the time step is chosen too large.

To make life easy, we shall settle for a first-order explicit time stepping, with a second-order discretization in space. Note that this needs both the left and the right neighbouring points, so it can only be used for the intervals except the first ($i = 1$) and last ($i = n$) one. For these we shall use the first order expressions (Eqns. 9 and 10).

So with the discretized equation of continuity one computes the density at every space point x_i at the new time $t + \Delta t$

$$\rho_i(t+\Delta t) = \rho_i - \Delta t \left(\frac{\rho_{i+1}(t)v_{i+1}(t) - \rho_{i-1}(t)v_{i-1}(t)}{x_{i+1} - x_{i-1}} \right) \tag{17}$$

From the momentum equation one computes the new velocities

$$\begin{aligned} v_i(t+\Delta t) &= \frac{(\rho v)_i(t+\Delta t)}{\rho_i(t+\Delta t)} \\ &= \frac{\rho_i(t)v_i(t)}{\rho_i(t+\Delta t)} - \frac{\Delta t}{\rho_i(t+\Delta t)}\left(\frac{\rho_{i+1}(t)(v_{i+1}^2(t)+c^2)-\rho_{i-1}(t)(v_{i-1}^2(t)+c^2)}{x_{i+1}-x_{i-1}}\right) \end{aligned} \tag{18}$$

In this formulation we chose to compute ρv at the new time, and then divide $\rho_i(t+\Delta t)v_i(t+\Delta t)$ by $\rho_i(t+\Delta t)$ to get the velocity. Another way would be to perform the factorial differentiation in Eqn. 7 and express $\partial\rho/\partial x$ from Eqn. 3. The particular choice may well influence the stability of the program, so either tests or analytical investigations of the formulation in question are often necessary to find out the method which gives the most accurate and stable results. Note that the above expressions must be modified at the first and last point as to accomodate the first-order terms (Eqns. 9 and 10).

Finally, we treat the energy equation by

$$e_i(t+\Delta t) = e_i(t+\Delta t) - \Delta t\left(\frac{(e_{i+1}(t)+p_i)v_{i+1}(t)-(e_{i-1}(t)+p_i)v_{i-1}(t)}{x_{i+1}-x_{i-1}}\right) \tag{19}$$

with the pressures $p_i = \rho_i(t)c_i^2(t)$. From the new value of e we compute the new value for $c_i^2(t+\Delta t) = 2(e_i(t+\Delta t)/\rho_i(t+\Delta t) - v_i^2(t+\Delta t))/3$ from the definition of e. Since $c^2 = kT/\mu$, this is taken as a measure of the temperature.

3.2 Treatment of the boundaries

So far, nothing has be said about how the boundaries of the computational region should be treated. The way we have formulated the discretization, we can see that any matter that flows across the left border ($x = 0$) with negative velocity will exit the region and is lost in all further computations; the same applies with matter flowing with positive velocity across $x = 1$. Thus, both boundaries are open.

We also have not specified that there is any matter flowing into the region. If we want to take into account that – for example – at $x = 0$ there is matter flowing in, we can do this by adding appropriate terms on the right hand sides of the equations for the first spatial grid point:

$$\rho_i(t+\Delta t) = \rho_i + \Delta t.... + \Delta t F_{\rm inner} \tag{20}$$

where $F_{\rm inner}$ is the mass flux rate at the inner boundary. In the momentum equation we add the momentum flux which is nothing but $F_{\rm inner}v_{\rm inner}$ with the velocity $v_{\rm inner}$ of the inflowing matter. Thus

$$(\rho v)_i(t+\Delta t) = \rho_i(t)v_i(t) - \Delta t\frac{\rho_{i+1}(t)(v_{i+1}^2(t)+c^2)-\rho_{i-1}(t)(v_{i-1}^2(t)+c^2)}{x_{i+1}-x_{i-1}} + \Delta t F_{\rm inner}v_{\rm inner} \tag{21}$$

from which we can compute the new velocity. For the energy equation we need to know the temperature (or the speed of sound $c_{\rm inner}$) of the inflowing material:

$$e_i(t+\Delta t) = e_i(t+\Delta t) - \Delta t\,(...) + \Delta t F_{\rm inner}c_{\rm inner}^2 \tag{22}$$

4 Program Structure

The program structure is very simple and straight-forward:

1. Initialize the basic constants, input the model parameters, initialize the initial conditions.
2. Compute the time changes of ρ and v for the first and last spatial grid points.
3. Compute the time changes of ρ and v for the grid points in between.
4. Compute the new values of ρ and v for all grid points.
5. Compute the smoothed values (see Sect. 6.4 below).
6. Check time. If still less than maximum time, increase time by Δt and go to Step (2). Otherwise end the program, print nice output.....

4.1 Which Time Step to Choose

In an hydrodynamical system, the changes of physical conditions, e.g. pressure, at one place — say x_1 — are propagated to another (x_2) by sound waves. Consider two neighbouring points: If a sound wave can travel within one time stepwidth Δt further than the distance between the points, the real physical system at x_2 will have completely responded to the change at x_1. But since the numerical treatment only neighbouring points may interact, the program cannot follow the changes properly. Thus a condition for a correct computation is that neighbouring points should be further away than the sound wave can reach in one time step

$$|x_1 - x_2| > c\Delta t \tag{23}$$

For any spatial grid spacing this give a maximum permissable time stepwidth (Courant Friedrichs Levy stepwidth)

$$\Delta t < \Delta t_{CFL} = \Delta x / c \tag{24}$$

To get accurate results, one should take Δt as small as practical, and certainly less than Δt_{CFL}. This condition is only valid for the first-order explicit scheme, and shows its limitations. If one wants to have a high spatial resolution to follow fine details, one **must** use correspondingly small time steps. Otherwise the numerical results are just nonsense.

In all of the following specific problems, one should make computations with several different time stepwidths and observe the consequences. From this experience one can select the stepwidth which is best for a particular purpose, either a fast-and-dirty overview of the evolution or a slow but accurate "production run".

5 How to proceed

1. first, develop a code to solve only the conservation of mass. Assume some initial density profile, and a constant velocity. Does your code shift this density profile across without changing its shape too much? An applet which does this, is available at
   `http://astro.u-strasbg.fr/~koppen/hydro/Advect.html`

2. Then add the momentum equation, so that you can compute the isothermal case, and check against the linear shock tube. (see below, or use the applet at `http://astro.u-strasbg.fr/~koppen/hydro/SimpleHydro.html`

3. chose one of the problems:

4. add energy equation and solve adiabatic shock tube

5. modify the code to spherical geometry and add gravity force to compute the isothermal collapse:
`http://astro.u-strasbg.fr/~koppen/hydro/Collapse.html`

6. energy equation and spherical geometry and heat conduction: evaporation of a gas could in a hot medium

7. write the code in the conservative formulation
`http://astro.u-strasbg.fr/~koppen/hydro/HydroSim.html` or/and the implicit formulation, ...

6 Some Tasks

6.1 The Isothermal Linear Shock Tube

Consider a tube with a fine membrane across its face, halfway between the two ends. The space on one side shall be filled with gas of some density ρ_1, the other space contains gas at a different density ρ_2. The gas in both regions shall be static, thus $v_i = 0$ at $t = 0$ for all points i. At time $t = 0$ the membrane bursts (instantly, of course), and the two gas volumes confront each other. Clearly, there should be a gas flow from the denser region into the dilute one, until the density is constant everywhere. This "simple" problem exhibits essential phenomena in hydrodynamics, such as a shock and a dilution wave. Yet it can be solved analytically – see standard textbooks on hydrodynamics – and therefore is used to check the stability and accuracy of a numerical code.

Numerical solutions of this problem can be obtained interactively by this Java Applet:
`http://astro.u-strasbg.fr/~koppen/hydro/HydroSim.html`

6.2 The Adiabatic Linear Shock Tube

The solution of the adiabatic problem is quite similar. In the density profile there are now two jumps, the right hand one being the shock. It is a very thin transition zone between gas of different densities. The exact (finite) thickness of real shocks are governed by microscopic physics, such as how fast atoms are excited and radiate away the energy. The other jump is the 'contact discontinuity' which separates the gas which came through the show from the gas of the initial high density zone. Steep changes in the solution are very difficult to treat numerically: if one wants to reproduce the fine details of the profiles, one must use a small time step, and thus long computing times. Our simple method is not very efficient. More sophisticated codes use an adaptive mesh for the grid points which

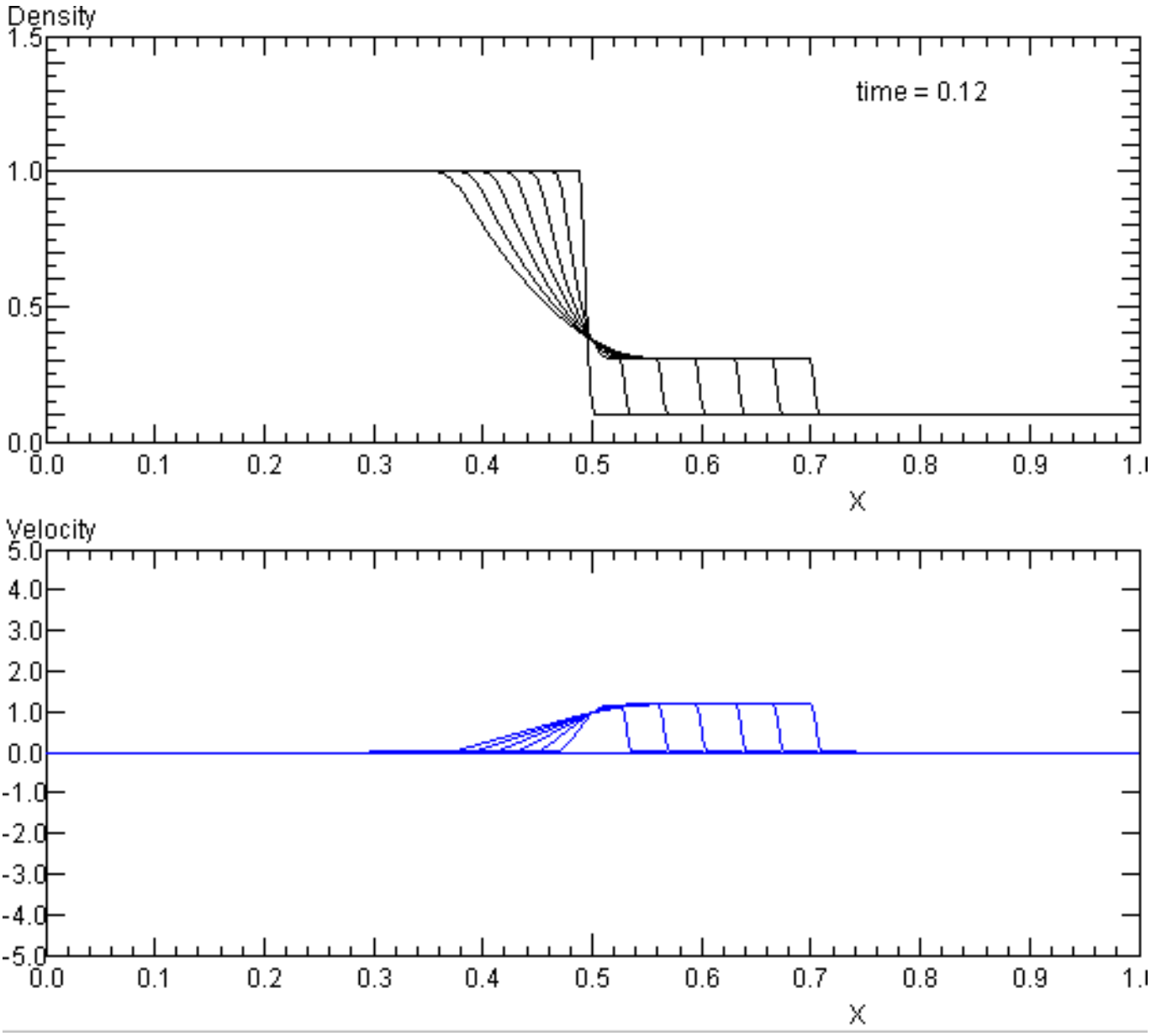

Figure 1: The isothermal linear shock tube. Shown are the profiles of density and velocity for the times 0, 0.02, 0.04, 0.06, 0.08, 0.10, and 0.12 time units after removal of the membrane. One time unit is the time in which a sound wave – speed 1.0 – crosses the computational domaine.

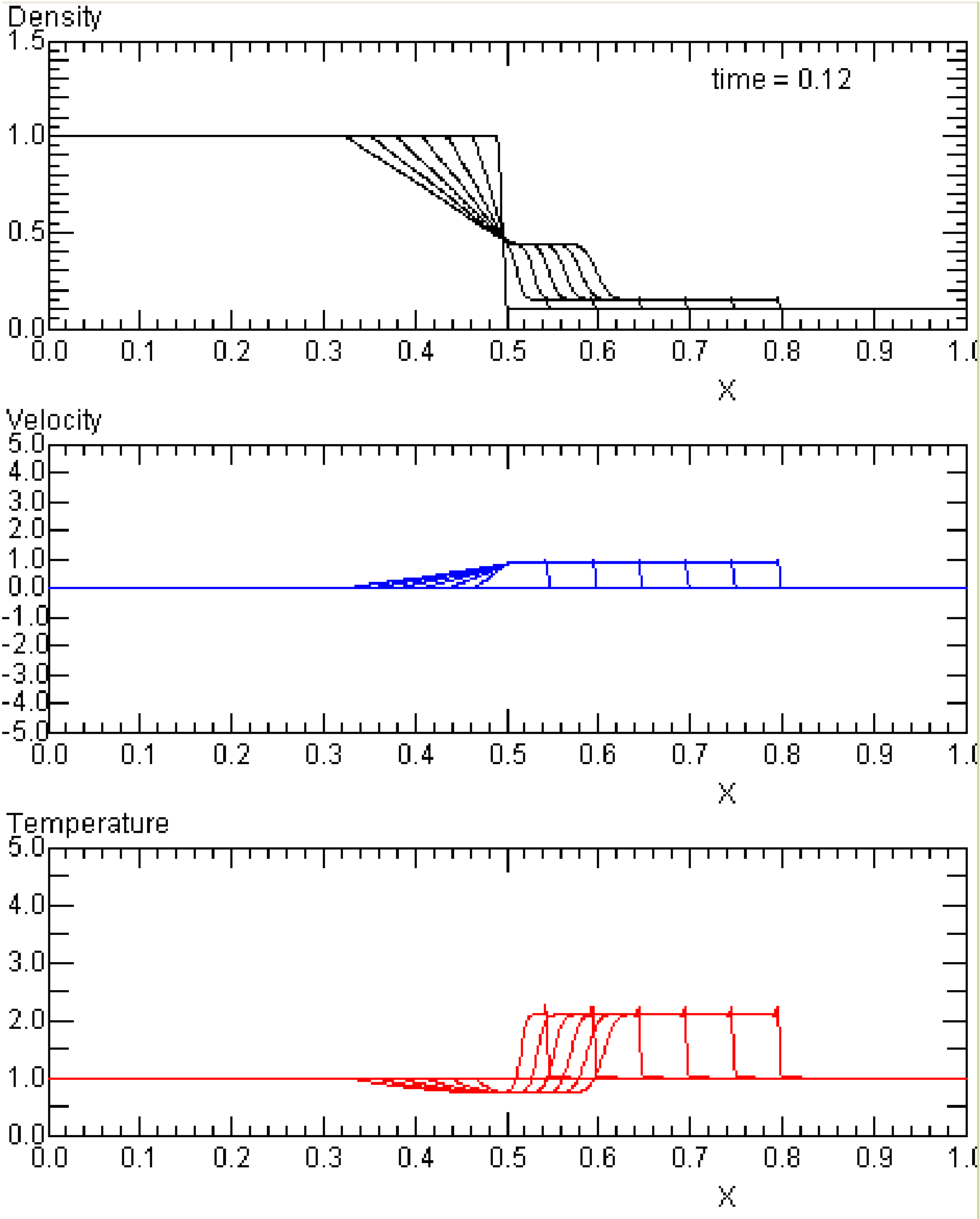

Figure 2: As above, but for the adiabatic case. Also shown are the profiles of the temperature. One unit temperature corresponds to a unit sound velocity.

are allowed to move where they are needed most, where the gradients are steep and next to jumps in the solution ...

6.3 What to expect

Our simple code can be expected to compute:

- the speed and the structure of the dilution wave. There is no sharp transition here, so you can expect to get good agreement with your code.
- the propagation of the shock: the position of the shock (i.e. the steepest part in the density profile) should move with the maximum value of the velocity.
- the speed of the shock: Bedijn and Tenorio-Tagle have shown that the speed v_s of an isothermal shock (sound speed c) is a function of the initial density contrast:

$$(\frac{v_s}{c})^2 \exp(\frac{v_s}{c} - \frac{c}{v_s}) = \frac{\rho_1}{\rho_2} > 1 \tag{25}$$

- Our program can verify this formula.

6.4 How to treat the shock

In the framework of only the hydrodynamical equations, the shock actually is a discontinuity. This poses a formidable problem for the numerical solution: When running your code, instabilities may develop at the shock and grow until the density profile is a mere zig-zag line, and negative densities occur. Don't dispair!

There are various ways how to tackle this problem: one might treat the shock as a real discontinuity, and follow the position of the shock. One may also add "artificial viscosity" terms to the equations, thereby spreading out the shock over a couple of grid points and effectively resolving it. A more simple way is to smear out the solution over a few grid points: At every timestep (or maybe every tenth) one applies a smoothing algorithm over all values and grid points:

$$y_i \rightarrow \hat{y}_i = \begin{cases} (1-2w)y_1 + 2wy_2 & \text{for } i = 1 \\ (1-2w)y_i + wy_{i-1} + wy_{i+1} & \text{for } 1 < i < n \\ (1-2w)y_n + 2wy_{n-1} & \text{for } i = n \end{cases} \tag{26}$$

The smoothing factor $w \approx 0.01$ is different for different situations, times, time steps, and how many time step you run the computation. There is no exact or optimum value. Simply find out by experiments: If it is too small, instabilities will grow, but if it is too large, the features of the density and velocity profiles are washed out too much.

6.5 Using Other Geometries: Spherically Symmetric Expansion of a Planetary Nebula or Nova Shell

The expansion of a nebular shell away from the central star is very similar to the linear shock tube problem. At time $t = 0$ a certain amount of gas is placed at some distance

from the star, maybe with an initial outflow velocity. Then expansion into interstellar space takes place.

There is one difference: The gas is moving radially away from the central star. Therefore we consider the spherically symmetric case, which can be taken into account by adding

$$\frac{2v_i}{x_i} \tag{27}$$

to the expression in the large bracket on the right hand side of Eqn. 17. Likewise, Eqn. 18 is modified by

$$\frac{2\rho_i v_i^2}{x_i} \tag{28}$$

Here we assume that the central star is at position $x = 0$. Obviously, taking $x = 0$ as one of the grid points, is inviting trouble, as both terms become infinitely large there. Also, as distances close to the star, these terms may become quite large anyway. Thus it is playing safe, if one avoids to compute the very early evolution close to the star.

If one wants to do a concrete calculation, one can distribute $1M_\odot$ of gas, over a spherical shell with inner radius 0.005 pc (1 parsec = $3\,10^{16}$m), which we set to the inner boundary x_{min} of the spatial grid, outer radius 0.03 pc. The initial velocity at all points should be 1 km/s, which is larger than the escape velocity at these distances, if the star was the sun. Thus one can neglect the influence of the gravitational field of the central star. For the outer boundary of the grid should be placed at e.g. 0.1 pc. Assume a temperature of 10^4K for all the gas, and a (number) density of the interstellar gas of 10^6 hydrogen atoms per cubic metre. The mass of the hydrogen atom is $1.66\,10^{-27}$kg.

6.6 Using Other Force Terms: Collapse of a Star or Galaxy

To compute the collapse of a gas cloud under the influence of its own gravitational field, we shall use the spherically symmetric case, as before, but in addition to the pressure gradient term in Eqn. 6 we also consider the gravitational force by the matter within the sphere of radius x, whose mass is M_x:

$$F_{grav} = -\frac{GM_x\rho}{x^2} = -\frac{G\rho}{x^2}4\pi\int_0^x \rho(r)r^2dr \tag{29}$$

Thus, the force at each radius is an integral over all points interior. If the spatial grid is sufficiently fine, one may evaluate the integral simply by the trapezoidal rule

$$\int_a^b f(x)dx \approx \frac{h}{2}(f(a)+f(b)) + h\sum_{i=2}^{n-1} f(x_i) \tag{30}$$

with $h = (b-a)/(n-1)$ and $x_i = a+h(i-1)$ or higher order schemes, such as Simpson's rule. Since the integral at one point x is part of the integral for any point exterior, it is well worth finding a economical way of summing up all the terms to give all the integrals needed.

- As before, one must avoid having $x = 0$ as part of the grid. Thus the region from $x = 0$ to the inner rim x_1 of the cloud is treated as a reservoir into which matter flowing across the inner boundary is collected. The mass of this core must also be taken into account when computing the gravitational force, of course.

- Initially, this core mass can be taken as

$$M_1(t = 0) = \frac{4\pi}{3} x_1^3 \rho_i$$

- During each time step this mass is increased by the flow

$$M_1(t + \Delta t) = M_1(t) + 4\pi x_1^2 v_1 \rho_1$$

- The way the forces are computed affects the stability of the solution, especially at the inner grid points, as the force changes strongly. Most probably, you will find many effects that seem to be upsetting the current understanding of the evolution of galaxies. However, as they will change if you change e.g. the spatial grid, they may well be due to inaccuracies, and you have to find ways to improve the program ... such as a non-linear spatial grid, extra-strong smoothing at preferentially inner points. Try it! To find tricks to make the program work accurately, is what one often spends quite a bit of time on.

6.7 The Free Fall Solution

A good test of the program is provided by considering the matter ignoring the thermal pressure (setting $c = 0$). Then the gas plunges towards the centre in a free fall. This can be solved analytically. Consider a homogeneous gas shell of radius r_0, density ρ_0, and velocity $v_0 = 0$ at time $t = 0$. At time t it will have collapsed to a radius r:

$$\frac{\pi}{2} \frac{t}{t_{ff}} = \sqrt{r r_0 - r^2} + r_0 \arcsin \sqrt{1 - r/r_0} \tag{31}$$

with the free fall timescale

$$t_{ff} = \sqrt{\frac{3\pi}{32 G \rho}} \tag{32}$$

In the early stages of the collapse ($t \ll t_{ff}$) one has

$$r/r_0 = 1 - \left(\frac{\pi}{4} \frac{t}{t_{ff}} \right)^2 \tag{33}$$

$$v = -\frac{\pi^2}{8} r_0 \frac{t}{t_{ff}^2} \tag{34}$$

i.e. the velocity increases linearly with distance from the centre, and it grows linearly with time. Since at that time all parts of a cloud are compressed in radius by a factor r_0/r, the initial density profile of the cloud will be preserved, and the density a a homogeneous

cloud is increased by a factor $(r_0/r)^3$.

At later stages ($t \gg 0$) we have $r \ll r_0$ and

$$r/r_0 = \left(\frac{t}{t_{ff}} - 1\right)^2 \tag{35}$$

$$v = \frac{2\sqrt{r r_0}}{t_{ff}} \tag{36}$$

This means that after one free fall time, the whole cloud has collapsed into the centre. However, it may be quite hard to compute the evolution that far. More likely, numerical instabilities have grown to produce negative densities before that time.

6.8 The Collapse of the Proto-Galaxy

The behaviour of a self gravitating gas cloud — irrespective of whether it will become a star or a galaxy — depends on whether it satisfies the criterion first formulated by James Jeans: A spherically symmetric and homogeneous gas cloud will collapse, if the gravitational energy exceeds the total thermal energy:

$$\frac{3}{5}\frac{GM^2}{R} > \frac{3}{2}NkT = \frac{3}{2}Mc^2 \tag{37}$$

This may also be interpreted as the escape velocity at the cloud's rim $v_{esc} = \sqrt{2GM/R}$ having to be $\sqrt{5} = 2.23$ times larger than the speed of sound, with which the cloud would disperse into intergalactic space. If it collapses, it does so initially with the free fall timescale; at later times, the pressure forces take over and slow down the collapse.

Compute the evolution of a protogalactic gas cloud of $10^{12} M_\odot$ of an initial radius of 30 kpc, with a gas temperature of 10^6 K. Does it satisfy the Jeans' criterion? Let the spatial grid cover radii from 1 kpc to 50 kpc, and place into the space from 30 to 50 kpc some thin gas, say $\rho_{outer} = 0.1\rho_{inner}$. What is going to happen? Now vary the mass of the cloud from a value lower than that satisfying Jeans' criterion to a value greater.

Turning to the formation of stars, can you say how the cloud could have looked like which had formed our Sun?

7 Extension: Conservative Formulation

Our simple approach has a serious disadvantage: it does not conserve mass, momentum, and energy strictly. You can check this by computing for each time step the total mass etc. which is present in the tube. Also, depending on how one formulates the boundary conditions, mass or energy can 'flow' across the left and right border and thus can be lost or gained numerically!

In the following we formulate the problem in a way that the code must conserve these quantities. The idea is this: think of the tube as made of a string of cells, with borders at

$x_i = 0, \Delta x, 2\Delta x, i\Delta x, ...1$, with the centers as $x_{ci} = 0.5\Delta x, ...$ In each cell, the density shall be constant $\rho_i = \rho x_{ci}$, so each cell contains the mass $m_i = \rho_i \Delta x A$ where A shall be the constant area of the tube's cross section. Integrating Eqn. 3 over each cell gives the equation for the change in mass:

$$\frac{dm_i}{t} = (Mv)|_{left} - (Mv)|_{right} \tag{38}$$

where Mv is the mass flux rate, and the terms show what mass is gained from flows (positive v means to the right) across the left border and how much is lost across the right one. To compute these rates, we shall use a simple recipe: Mv shall be the density of the cell where the flow comes from multiplied with the velocity at the border (NOT at the cell centre). So we have for the first term

$$(Mv)|_{left} = \begin{cases} m_{i-1} \cdot v_{i-\frac{1}{2}} & \text{for } v_{i-\frac{1}{2}} > 0 \\ m_i \cdot v_{i-\frac{1}{2}} & \text{for } v_{i-\frac{1}{2}} < 0 \end{cases} \tag{39}$$

so depending on the sign of the velocity $v_{i-\frac{1}{2}}$ at the cell border, the cell i either gets matter from the left cell $(i-1)$ or looses it to the right neighbour $(i+1)$.

Note that with this formulation, one can easily put in the boundary conditions by specifying the respective mass flow rates.

How to compute the velocities at the cells' borders? Let us use a simple recipe $v_{i-\frac{1}{2}} = (v_{i-1} + v_i)/2$ which relates them to the values at the cell centers.

In this fashion, we solve this equation by simple time-stepping, similar as before.

Now we treat the momentum equation in an analoguous way, and get the equation for the momentum in cell i which is simply $(Mv)_i = M_i v_i$, as v_i was meant to refer to the cell centre:

$$\frac{d(Mv)_i}{t} = (Mvv)|_{left} - (Mvv)|_{right} + (p)|_{right} - (p)|_{left} \tag{40}$$

Here we have the rates of momentum flowing into/out of the cell which shall be computed as:

$$(Mvv)|_{left} = \begin{cases} (m_{i-1} v_{i-1}) \cdot v_{i-\frac{1}{2}} & \text{for } v_{i-\frac{1}{2}} > 0 \\ (m_i v_i) \cdot v_{i-\frac{1}{2}} & \text{for } v_{i-\frac{1}{2}} < 0 \end{cases} \tag{41}$$

But we also have to compute the pressure gradient across the cell, which we shall do like this

$$(p)|_{right} = \rho_{i+1} c_{i+1}^2 \tag{42}$$

Note that in principle we have to compute pA, but as A is constant, I've dropped it in the formulae.

The procedure is like this: in each time step

- compute the new values for M_i and $(Mv)_i$
- get the new velocities $v_i = (Mv)_i / M_i$

- get the new border velocities $v_{i\pm\frac{1}{2}}$

And now the same thing for the equation of the energy $E_i = e_i \Delta x A$ contained in cell i:

$$\frac{dE}{dt} = ((E+p)v)|_{left} - ((E+p)v)|_{right} \tag{43}$$

which is done with

$$((E+p)v)|_{left} = \begin{cases} (E_{i-1}+p_{i-1}) \cdot v_{i-\frac{1}{2}} & \text{for } v_{i-\frac{1}{2}} > 0 \\ (E_i + p_i) \cdot v_{i-\frac{1}{2}} & \text{for } v_{i-\frac{1}{2}} < 0 \end{cases} \tag{44}$$

with

$$E_i + p_i = \rho_i (\frac{5}{2} c_i^2 + \frac{1}{2} v_i^2) \Delta x \tag{45}$$

and we proceed as before.

8 Extension: Implicit Formulation

The explicit time-stepping is nice and simple, but it has one disadvantage: the time step must be chosen sufficiently small, so that any information (which propagates with the speed of sound or the flow velocity) moves from one spatial grid point to the next sufficiently slowly, say it takes 10 time step to do that. Otherwise, numerical instabilities can easily form and grow, giving completely false results. This means that high spatial resolution necessitates a small time step, i.e. a long execution time.

The implicit formulation is more stable, and allows larger time steps. Let us apply it to the conservation of mass

$$m_i(t+\Delta t) = m_i(t) - \Delta t \left((Mv)|_{left} - (Mv)|_{right} \right) \tag{46}$$

Note that in constrast to Eqn. (22), all quantities of the right hand side are to be taken at the time $t + \Delta t$!! If the velocities at the cell boundaries are positive,

$$m_i(t+\Delta t) = m_i(t) - \Delta t \left(m_{i-1}(t+\Delta t) \cdot v_{i-\frac{1}{2}}(t+\Delta t) - m_i(t+\Delta t) \cdot v_{i+\frac{1}{2}}(t+\Delta t) \right) \tag{47}$$

If the velocities were fixed, the Eqns. (30), written down for all spatial points i would form a system of linear equations for the unknown new cell masses $m_i(t+\Delta t)$, given the old cell masses $m_i(t)$.

$$m_{i-1}(t+\Delta t)(\Delta t v_{i-\frac{1}{2}}(t+\Delta t)) + m_i(t+\Delta t) \left(1 + v_{i+\frac{1}{2}}(t+\Delta t) \right) = m_i(t) \tag{48}$$

If we include the cases for negative flow velocities, we note that we get a system of tridiagonal structure:

$$m_{i-1}(t+\Delta t) A_{i-1,i} + m_i(t+\Delta t) A_{i,i} + m_{i+1}(t+\Delta t) A_{i+1,i} = m_i(t) \tag{49}$$

All coefficients of the matrix $A_{i,k}$ except the diagonal and its two neighbours are zero. One solves this system by a Gaussian elimination but limited only to first removing the

coefficients below the diagonal (going from $i = 1$ to n), and then moving backwards, resolving the simple equations of the type $m_i a_{i,i} + m_{i+1} a_{i,i+1} = b_i$.

Of course, the velocities are not fixed! And they are computed from the conservation of momentum. So we apply the same technique to that equation, and get now a (bigger) system of equations for the new cell masses and velocities. Unfortunately, the equations are no longer linear: the coefficients A of the mass equations depend on the velocities. There are techniques to resolve such systems, like the Newton-Raphson iteration. The program can become quite complex, and whenever one wants to change the equations by including other physical processes, it might mean a lot of work.

But let us try out something more simple: **Operator splitting**. If we solve the mass equations, assuming that the velocities are the old velocities, we have to solve only linear equations. Knowing the new masses, we can solve the momentum equations (which would be linear in the Mvs) also by linear inversion. And finally, we solve the energy equations. I do not give here all the resulting equations in detail, you can do this?

Now we have two possibilities: (a) we proceed directly to the next time step, starting with the mass equations etc.... Surely, this is but an approximation, since solving each set of equations, we had assumed that the other quantities are constant. But may be this is accurate enough. (b) The other way is to iterate the procedure until all masses, all velocities, and all thermal energies are stable for this new time point: For example, after solving mass and momentum equations, we take the new velocities to re-compute the masses via Eqn.(30), then we re-solve the momentum equations etc... Finally when masses and velocities are sufficiently accurate, we solve the energy equation. Since the energy equation determines the speed of sound, which is used in the momentum equations, we then should re-solve the momentum equation, ... It depends on the specific problem and the accuracy one needs, which strategy of iteration one might use, or whether one does not need to do iterations for certain equations... You need to experiment here and carefully check what comes out.